NASA TM-86021

NASA Technical Memorandum 86021

Development of Computational Fluid Dynamics at NASA Ames Research Center

Mamoru Inouye

October 1984

National Aeronautics and
Space Administration

NASA Technical Memorandum 86021

Development of Computational Fluid Dynamics at NASA Ames Research Center

Mamoru Inouye, Ames Research Center, Moffett Field, California

National Aeronautics and
Space Administration

Ames Research Center
Moffett Field, California 94035

N85-12 866 #

DEVELOPMENT OF COMPUTATIONAL FLUID DYNAMICS

AT NASA AMES RESEARCH CENTER

Mamoru Inouye

Ames Research Center

SUMMARY

Ames Research Center has the lead role among NASA centers to conduct research in computational fluid dynamics. The past, the present, and the future prospects in this field are reviewed. Past accomplishments include pioneering computer simulations of fluid dynamics problems that have made computers valuable in complementing wind tunnels for aerodynamics research. The present facilities include the most powerful computers built in the United States. Three examples of viscous flow simulations are presented: an afterbody with an exhaust plume, a blunt fin mounted on a flat plate, and the Space Shuttle. The future prospects include implementation of the Numerical Aerodynamic Simulation Processing System that will provide the capability for solving the viscous flow field around an aircraft in a matter of minutes.

INTRODUCTION

A major goal of the National Aeronautics and Space Administration (NASA) is to conduct an effective and productive aeronautics research and technology program that contributes materially to civil and military aviation. A key objective in attaining this goal is the advancement of both the understanding of and the capability to predict aerodynamic phenomena through the use of wind tunnels and high-speed computers. The results of past efforts are exemplified by the large number of commercial airliners built in the United States and operated worldwide, including Japan (fig. 1).

In support of NASA's goal, Ames Research Center has established and strives to maintain one of the world's premier aeronautical research establishments. In computational fluid dynamics, Ames Research Center has the lead role among NASA centers and has

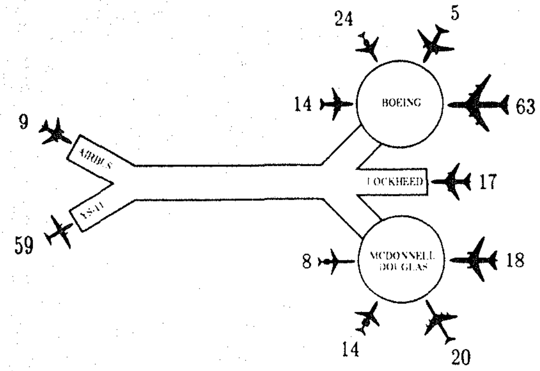

Figure 1.- Japanese commercial airliners in 1983 (number in service).

installed the most powerful computers built in the United States. The research program encompasses the development of numerical methods and computer codes for fluid-flow phenomena, experiments for verifying the codes, and application of the codes to practical aerodynamics problems. This paper reviews the past, the present, and the future prospects of computational fluid dynamics, based on experiences at Ames Research Center.

THE PAST

The history of computational fluid dynamics at Ames Research Center began 25 years ago with the establishment of NASA when Ames Aeronautical Laboratory was transferred from the National Advisory Committee for Aeronautics (NACA). Van Dyke (ref. 1) used an IBM 650 computer to obtain numerical solutions of the inviscid flow field around a blunt body traveling at supersonic speeds. Reentry into Earth's atmosphere by space capsules and missiles was the application of such studies. This era also saw the beginning of the commercial jet age, with transcontinental and transatlantic flights by Boeing 707 and Douglas DC-8 airliners.

Fifteen years ago, computational fluid dynamics became recognized as a separate field of study. The combination of the

availability of the IBM 360 series computers and development of new algorithms allowed solution of problems previously considered infeasible. MacCormack (ref. 2) devised an algorithm for the Navier-Stokes equations and solved numerically the interaction between a shock wave and a laminar boundary layer; several investigators obtained solutions of the small-disturbance equations for inviscid transonic flow over an airfoil. These results were reported at the Second International Conference on Numerical Methods in Fluid Dynamics held at Berkeley, California, September 15-19, 1970 (ref. 3). Air travel was augmented by the introduction of the jumbo Boeing 747, now the standard airliner for intercontinental flights.

Ten years ago, transonic flow solutions were obtained for inviscid flow over wings by Lomax et al. (ref. 4) and for viscous flow over airfoils by Deiwert (ref. 5). The Control Data Corp. (CDC) 7600 became available for fluid dynamics simulations, and the ILLIAC IV, a parallel processing computer with 64 separate processing elements, was installed at Ames. The ILLIAC IV computer was used extensively to study turbulence from first principles, beginning with the direct numerical simulations by Rogallo (ref. 6). A major event in air travel was the introduction of the British-French Concorde supersonic airliner.

Five years ago, unsteady viscous flow solutions were obtained for the aileron buzz problem by Steger and Bailey (ref. 7), and steady three-dimensional viscous flow solutions were obtained for axisymmetric bodies at angle of attack by Pulliam and Steger (ref. 8) and by Hung (ref. 9). In a program conducted by Stanford University and Ames Research Center, calculations of turbulence were performed on the ILLIAC IV computer, using the large-eddy simulation technique, in which large-scale eddies are calculated from the unsteady Navier-Stokes equations and subgrid-scale eddies are modeled. Results for turbulent channel flow were obtained by Moin and Kim (ref. 10). During the past 5 years, the solution of three-dimensional viscous flow problems has become the major focus of interest.

THE PRESENT

The status of computational fluid dynamics at Ames Research Center is that of a maturing science that complements the role of wind tunnels in aerodynamics research. The computer facilities will be described first, followed by a short discussion of numerical methods and by the presentation of three examples of viscous flow simulations: an afterbody with an exhaust plume, a blunt fin mounted on a flat plate, and the Space Shuttle.

Computer Facilities

The computers used primarily for computational fluid dynamics research at Ames Research Center include Cray X-MP/22 and CDC Cyber 205 supercomputers and several VAX-11/780 minicomputers. All the computers are linked by high-speed lines so that data can be transferred between any nodes in the network as shown in figure 2.

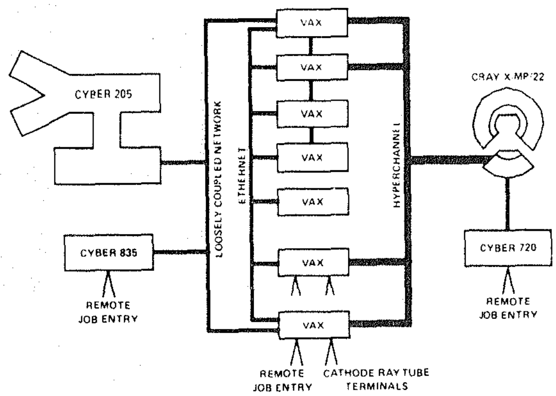

Figure 2.- Ames Research Center supercomputer network.

The Cray X-MP supercomputer has two processors and storage totaling 2 million 64-bit words of main memory, 16 million words of extended solid-state memory, and 1.2 billion words on disks. The Cray is operated in a batch mode, with jobs submitted through the Cyber 720 front-end processor and through four VAX systems connected by Hyperchannel links. The current Cray operating system does not allow multitasking, but a future version will; multitasking allows a job to be executed on both processors.

The Cyber 205 supercomputer has four pipelines and storage totaling 4 million 64-bit words of main memory and 1.2 billion words on disk. This computer is also operated in a batch mode, with jobs submitted through the Cyber 835 front-end processor and through two VAX systems connected through a loosely coupled network.

A typical VAX-11/780 system connected directly to the supercomputers has 8 megabytes of main memory and in excess of 600 megabytes of disk storage. Individual researchers work interactively with the VAX system from a cathode-ray-tube terminal to edit programs and submit jobs to either the Cray X-MP/22 or Cyber 205 supercomputer. The results are processed on the VAX system and may be displayed on the cathode-ray-tube terminal, as shown in figure 3.

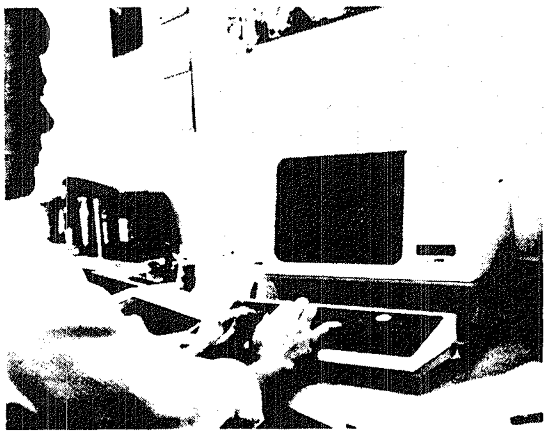

Figure 3.- Cathode-ray-tube terminal used to access Ames computer facilities.

Numerical Methods

The levels of approximation for the governing equations solved in current computational fluid dynamics studies are shown in figure 4. Early work solved the inviscid equations, beginning with linearized theory, followed by the small-disturbance and full-potential equations, and, most recently, the Euler equations. Viscous effects are solved separately using the boundary-layer equations, which may or may not be coupled with the inviscid equations. This approach is valid for high-Reynolds-number flow with little or no flow separation.

Recent work has concentrated on the

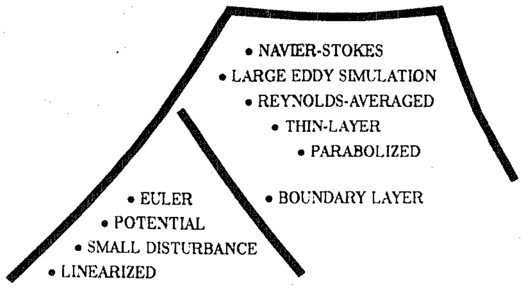

Figure 4.- Levels of approximation for fluid dynamics equations.

Reynolds-averaged Navier-Stokes equations, with the parabolized approximation allowing marching in the streamwise direction for supersonic flow and the thin-layer approximation neglecting viscous diffusion along the surface. Turbulence modeling is required in all these studies, with the Baldwin-Lomax model (ref. 11) being the one most commonly used. The large-eddy simulation method requires only that the subgrid-scale turbulence be modeled. The ultimate goal is to solve the complete, unsteady Navier-Stokes equations without any turbulence modeling for complex configurations. In general, however, as the level of approximation rises, the geometry must be less complex because of limitations on computer speed and storage.

The selection of the grid is an important step. Some of the criteria to be met are that the grid should conform to the body, be clustered where flow gradients are steep, match the flow physics, simplify application of boundary conditions, and produce well-ordered equations adaptable to vector processing on the latest supercomputers. The grid-generation methods require solution of either algebraic or partial differential equations; both are in common use.

Originally, the most popular algorithm for the solution of the Navier-Stokes equations was the MacCormack method (ref. 2), which was explicit in time and used central differencing in space. Considerable computer time was required, especially for high-Reynolds-number flows, because the stability condition for explicit methods allows only a small time-step for a small mesh. Implicit methods such as the Beam-Warming method (ref. 12), which allow much larger time-steps, were developed as a result and are

widely used today. Implicit methods, however, require more computer time per timestep and generally are more difficult to program. Recently, MacCormack (ref. 13) modified his original method to make it implicit and more efficient by several orders of magnitude.

Example 1: Afterbody with an Exhaust Plume

The first example of a viscous flow simulation is for an afterbody with an exhaust plane. Significant drag can result from the interaction between the exhaust and the flow field surrounding a missile, rocket, aircraft afterbody, or nacelle. Solutions of the thin-layer form of the Navier-Stokes equations were obtained by Deiwert (ref. 14) for flow over an axisymmetric afterbody containing a centered propulsive jet. Nakahashi and Deiwert (ref. 15) have recently improved the efficiency and accuracy of the solution method by introducing an adaptive grid, which clusters grid points in the vicinity of shock waves and slip lines where the density gradients are large. The original grid and the final solution-adaptive grid are shown in figure 5(a) for a free-stream Mach number of 2.0, a jet exit Mach number of 2.5, a Reynolds number based on body diameter of 0.65×10^6, and an exhaust jet to free-stream static pressure ratio of 6.0. The density contours obtained with the fixed original grid and the adaptive grid are shown in figure 5(b). Significant improvements in the solution are noted in the vicinity of the compression shock, barrel shock, and Mach disk.

Example 2: Blunt Fin Mounted on a Flat Plate

The second example of a viscous flow simulation is that of a blunt fin mounted on a flat plate. Loss of control effectiveness and severe local heating are some of the consequences of supersonic flow over bodies in regions where wings or fins are attached. The bow shock wave ahead of the blunt fin interacts with the boundary layer on the flat plate (or body) and causes separation of the flow as shown in figure 6. Solutions of the thin-layer form of the Navier-Stokes equations were obtained by Hung and Buning (ref. 16) for a free-stream Mach number of 2.95, a unit Reynolds number of 63×10^6/m, and a fin diameter of 1.27 cm. Particle paths are shown in figure 7 near the plane of symmetry for the separated region. Particles near the flat plate separate from the surface, reverse direction, and spiral out past the blunt fin. Particles away from

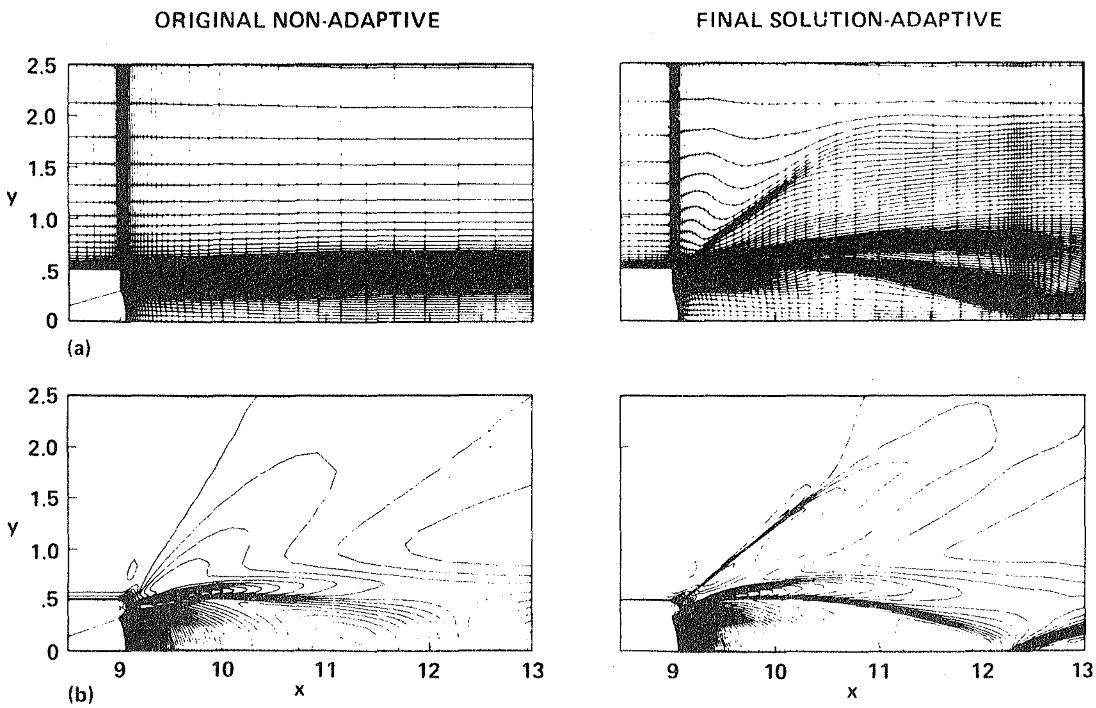

Figure 5.- Afterbody flow field with exhaust plume. (a) Grid; (b) density contours.

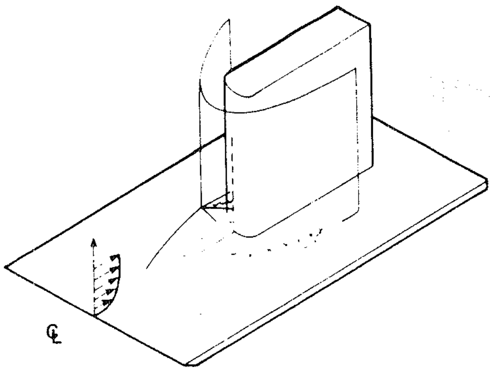

Figure 6.- Blunt fin mounted on flat plate.

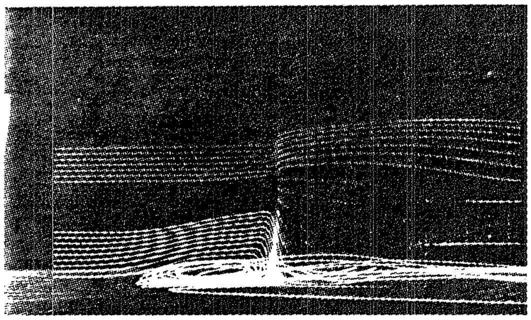

Figure 7.- Particle paths near plane of symmetry of separated region ahead of blunt fin.

the flat plate are deflected around the blunt fin.

Example 3: Space Shuttle

The last example of a viscous flow simulation is for the Space Shuttle. During reentry into the atmosphere at high angle of attack, intense heating occurs on both the windward surface and regions of the leeward surface where flow reattaches after separation. Solutions of the parabolized Navier-Stokes equations were obtained by Chaussee et al. (ref. 17) for the flow field around the forward two-thirds of the Space Shuttle. Simulated oil-flow lines for turbulent flow on the surface are shown in figure 8 for the following wind-tunnel conditions: a free-stream Mach number of 7.9, a Reynolds number of 1.37×10^6 based on vehicle length, and an angle of attack of 25°. Separation is indicated on the strake-wing by the coalescence of oil-flow lines, followed by reattachment part way up the fuselage, and separation again on the upper surface.

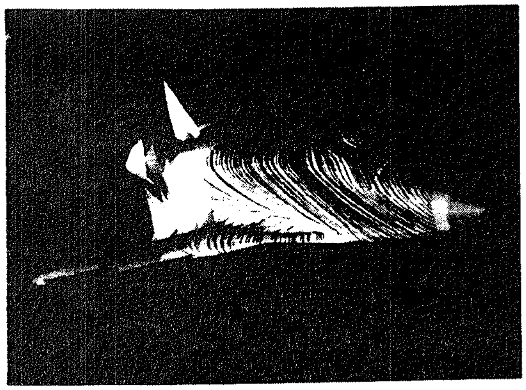

Figure 8.- Simulated oil flow lines on Space Shuttle surface.

THE FUTURE

To ensure continuing leadership in computational fluid dynamics and related disciplines, NASA has initiated the Numerical Aerodynamic Simulation (NAS) Program to establish a national computational facility at Ames Research Center. The NAS Processing System will comprise six subsystems: high-speed processor, mass storage, support processing, long-haul communication, graphics, and work station. All six are connected by the data network subsystem, as shown in figure 9.

The heart of the system will be the high-speed processor subsystem, which will employ the fastest processors available at any given time. The initial configuration, which will be operational in 1986, will have a Cray-2, with a sustained speed of 250 million floating-point operations per second (mflops), a main memory of 64 million 64-bit words, and 15.6 gigabytes of disk storage. In 1988, this processor will be supplemented

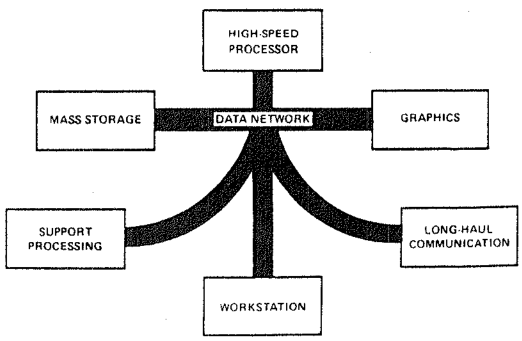

Figure 9.- Numerical aerodynamic simulation processing system.

by one expected to have a sustained speed of 1 billion floating-point operations per second (gflops) and a main memory of 256 million words. The mass storage subsystem will provide on-line and archival storage with an initial capacity of 200 gigabytes. The support processing subsystem will provide interactive support for remote users and input/output devices, such as a printer, film recorder, and tape drives. The long-haul communication subsystem will provide access to the NAS Processing System from remote sites, including other government laboratories, universities, and industry. The graphics subsystem will provide hard-copy graphical output for all users and dynamic high-resolution color displays for on-site users. The work-station subsystem will provide access to the NAS Processing System for on-site research scientists and engineers. The initial configuration will have 25 self-contained units, each equipped with a processor, color graphics display, dot matrix printer, and 500 megabytes of disk storage to develop computer codes, process output data from the high-speed processor, and prepare reports.

Successful implementation of the NAS Processing System in 1988 will provide the capability of solving the Reynolds-averaged Navier-Stokes equations for an aircraft configuration in a matter of minutes.

CONCLUDING REMARKS

Continuing progress in computational fluid dynamics will require research and development in turbulence modeling, numerical methods, and computer hardware.

The large range of length scales associated with turbulent flow precludes solution of the unsteady Navier-Stokes equations for complex configurations with the computers that will be available in the near future. Turbulence modeling will be required to solve the Reynolds-averaged Navier-Stokes equations for three-dimensional flows with separation. Both wind-tunnel and computer experiments are needed. New experimental techniques using nonintrusive laser devices will be complemented by computer simulations of turbulence.

Numerical methods have become more efficient during the past 15 years, but more gains are possible. The need for hundreds and, sometimes, as many as thousands of iterations to solve a steady flow problem is not acceptable. The use of implicit methods will avoid the time-step restrictions for stability inherent in explicit methods.

Advances in computer architecture and semiconductor technology are resulting in supercomputers that are faster and that have larger memories (fig. 10). Increasing the number of processors is a straightforward way to increase the speed, and the processors may be assigned different tasks for the same problem. The numerical method must take into consideration the computer architecture in order to optimize the use of the available hardware.

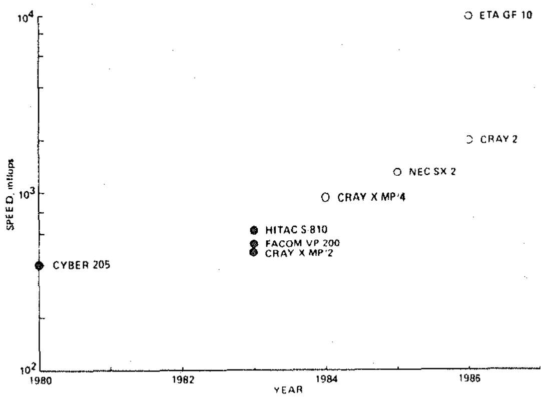

Figure 10.- Peak processing speeds for current and future supercomputers.

REFERENCES

1. Van Dyke, M. D.: The Supersonic Blunt-Body Problem--Review and Extension. J. Aerosp. Sci., vol. 25, no. 8, Aug. 1958, pp. 485-496.
2. MacCormack, R. W.: The Effect of Viscosity in Hypervelocity Impact Cratering. AIAA Paper 69-354, Cincinnati, Ohio, 1969.
3. Holt, M., ed.: Proceedings of the Second International Conference on Numerical Methods in Fluid Dynamics. Lecture Notes in Physics, vol. 8, Springer-Verlag, Berlin, 1971.

4. Lomax, H.; Bailey, F. R.; and Ballhaus, W. F.: On the Numerical Simulation of Three-Dimensional Flow with Application to the C-141 Wing. NASA TN D-6933, 1973.
5. Deiwert, G. S.: Numerical Simulation of High Reynolds Number Transonic Flows. AIAA Paper 74-603, Palo Alto, Calif., 1974. (Also AIAA J., vol. 13, no. 10, Oct. 1975, pp. 1354-1359.)
6. Rogallo, R. S.: An ILLIAC Program for the Numerical Simulation of Homogeneous Incompressible Turbulence. NASA TM-73203, 1977.
7. Steger, J. L.; and Bailey, H. E.: Calculation of Transonic Aileron Buzz. AIAA Paper 79-0134, New Orleans, La., 1979. (Also AIAA J., vol. 18, no. 3, Mar. 1980, pp. 249-255.)
8. Pulliam, T. H.; and Steger, J. L.: Implicit Finite-Difference Simulations of Three-Dimensional Compressible Flow. AIAA Paper 78-10, Huntsville, Ala., 1978. (Also AIAA J., vol. 18, no. 2, Feb. 1980, pp. 159-167.)
9. Hung, C. M.: Numerical Solution of Supersonic Laminar Flow over an Inclined Body of Revolution. AIAA Paper 79-1547, Williamsburg, Va., 1979. (Also AIAA J., vol. 18, no. 8, Aug. 1980, pp. 921-928.)
10. Moin, P.; and Kim, J.: Numerical Investigation of Turbulent Channel Flow. J. Fluid Mech., vol. 18, May 1982, pp. 341-377.
11. Baldwin, B. S.; and Lomax, H.: Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows. AIAA Paper 78-257, Huntsville, Ala., 1978.
12. Beam, R. M.; and Warming, R. F.: An Implicit Factored Scheme for the Compressible Navier-Stokes Equations. AIAA Paper 77-645, Albuquerque, N. Mex., 1977. (Also AIAA J., vol. 16, no. 4, Apr. 1978, pp. 393-402.)
13. MacCormack, R. W.: A Numerical Method for Solving the Equations of Compressible Viscous Flow. AIAA Paper 81-0110, St. Louis, Mo., 1981. (Also AIAA J., vol. 20, no. 9, Sept. 1982, pp. 1275-1281.)
14. Deiwert, G. S.: A Computational Investigation of Supersonic Axisymmetric Flow over Boattails Containing a Centered Propulsive Jet. AIAA Paper 83-0462, Reno, Nev., 1983.
15. Nakahashi, K.; and Deiwert, G. S.: A Practical Adaptive Grid Method for Complex Fluid Flow Problems. Ninth International Conference on Numerical Methods in Fluid Dynamics, Saclay, France, June 25-29, 1984.
16. Hung, C. M.; and Buning, P. G.: Simulation of Blunt-Fin Induced Shock Wave and Turbulent Boundary Layer Interaction. AIAA Paper 84-0457, Reno, Nev., 1984.
17. Chaussee, D. S.; Rizk, Y. M.; and Buning, P. G.: Viscous Computation of a Space Shuttle Flow Field. Ninth International Conference on Numerical Methods in Fluid Dynamics, Saclay, France, June 25-29, 1984.

1. Report No. NASA TM-86021	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle DEVELOPMENT OF COMPUTATIONAL FLUID DYNAMICS AT NASA AMES RESEARCH CENTER		5. Report Date November 1984
		6. Performing Organization Code
7. Author(s) Mamoru Inouye		8. Performing Organization Report No. A-9887
9. Performing Organization Name and Address Ames Research Center Moffett Field, Calif. 94035		10. Work Unit No.
		11. Contract or Grant No.
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, D.C. 20546		13. Type of Report and Period Covered Technical Memorandum
		14. Sponsoring Agency Code 505-31-01

15. Supplementary Notes

Point of Contact: Mamoru Inouye, Ames Research Center, MS 202A-1, Moffett Field, Calif. 94035 (415) 965-5126 or FTS 448-5126

16. Abstract

Ames Research Center has the lead role among NASA centers to conduct research in computational fluid dynamics. The past, the present, and the future prospects in this field are reviewed. Past accomplishments include pioneering computer simulations of fluid dynamics problems that have made computers valuable in complementing wind tunnels for aerodynamic research. The present facilities include the most powerful computers built in the United States. Three examples of viscous flow simulations are presented: an afterbody with an exhaust plume, a blunt fin mounted on a flat plate, and the Space Shuttle. The future prospects include implementation of the Numerical Aerodynamic Simulation Processing System that will provide the capability for solving the viscous flow field around an aircraft in a matter of minutes.

17. Key Words (Suggested by Author(s)) Computers Aerodynamics Computational fluid dynamics		18. Distribution Statement Unlimited Subject Category - 02	
19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of Pages 10	22. Price* A02

*For sale by the National Technical Information Service, Springfield, Virginia 22161

End of Document